GLOBAL JOURNAL OF ENGINEERING, DESIGN & TECHNOLOGY
(Published By: Global Institute for Research & Education)
www.gifre.org

COMPUTATIONAL ANALYSIS OF FLOW BEHAVIOUR IN CURVED SECTION OF THE SERPENTINE INTAKE

P Anand[1], D Lakshmanan[2] & R Saravanan[3]
[1,2]Department of Aeronautical Engineering, Bannari Amman institute of technology, Sathyamangalam, India
[3]Department of Aeronautical department, Nehru institute of engineering and technology, India

ABSTRACT

The purpose of this project is to understand the flow deflection in the curved section of the serpentine intake using CFD analysis. Serpentine intake is the new generation intake used to increase the stealth property of an aircraft. It is used to eliminate the moving parts in the jet engine intake. Varying the shape of the inlet duct reduces the weight. Normally intake spikes are used to produce shock hence increase the pressure of the incoming flow. In serpentine intake there is no need of spike to reflect the flow. It consists of two curved sections in its body and deflects the flow which forms shock thus it reduces the velocity of the incoming flow. By varying the curvature ratio between the sections of the serpentine intake we can achieve significant pressure change due to the changes in the deflection angle of the flow.

Keywords: *serpentine intake, radar cross section, CFX solver, shock waves*

1. INTRODUCTION

1.1 Intake:

Air intake is one of the most essential parts of a jet propulsion system in an air vehicle. It has the responsibility of guiding the uniform free stream through the ducts and delivering it with the most effective and suitable velocity distribution to the engine face while converting kinetic energy to static pressure.There are two main categories of air intakes: subsonic and supersonic. On both flight conditions, each air intake has particular characteristics that would be altered it to act properly in that flight regime.

Geometrically, there will be various shapes of air intakes exist: Straight, “S-shaped”, Serpentine and, etc. The geometry and characteristic of air intake depend significantly on the mission, and designer provisions.

For subsonic aircraft, the inlet is a duct which is required to ensure smooth airflow into the engine despite air approaching the inlet from directions other than straight ahead. This occurs on the ground from cross winds and in flight with aircraft pitch and yaw motions.

The duct length is minimised to reduce drag and weight. Air enters the compressor at about half the speed of sound so at flight speeds lower than this the flow will accelerate along the inlet and at higher flight speeds it will slow down.

Thus the internal profile of the inlet has to accommodate both accelerating and diffusing flow without undue losses. For supersonic aircraft, the inlet has features such as cones and ramps to produce the most efficient series of shockwaves which form when supersonic flow slows down. The air slows down from the flight speed to subsonic velocity through the shockwaves, then to about half the speed of sound at the compressor through the subsonic part of the inlet.

The particular system of shockwaves is chosen, with regard to many constraints such as cost and operational needs, to minimise losses which in turn maximises the pressure recovery at the compressor.

1.2 Serpentine Intake:

As aircraft continue to evolve, the requirements for their development are becoming more and more stringent. For military applications, the overall thrust to weight ratio requirement is increasing, demanding either lighter aircraft through length reductions or lighter materials, or more powerful engines which tend to add weight reductions or lighter materials, or more powerful engines which tend to add weight.

By reducing the overall aircraft length, large weight savings may be realized with an increase in overall thrust to weight ratio for the same engines. A reduction in weight leads to lower costs and lighter, more compact high performance aircraft. Reduction in radar cross-section is another area of design focus. With advancements in radar technology, it is necessary to continue to decrease the radar cross-section (RCS) of aircraft.

Many technologies have been developed which decrease the radar cross section of an aircraft. A large reduction in RCS can be realized by eliminating the engine fan face as a radar return source. The fan face can be hidden from radar by developing an inlet duct that is offset so that there is no direct line of sight from the entrance of the inlet duct to the engine fan face.

This prevents a direct path for radar beams to strike the engine fan face and return to the receiver. These ducts are included in a general class of serpentine inlet ducts. Serpentine inlet ducts have been in existence for some time and can be found in many of today’s commercial and military aircraft including the Boeing 727, F-16 and F-117. Commercial aircraft use serpentine ducts to allow the thrust vector from rear mounted engines to be aligned with the axis of the aircraft.

To improve aircraft thrust to weight ratio, the serpentine shape of the inlet duct is becoming more aggressive as an ultra-compact, highly offset inlet duct, allowing a reduction in the overall aircraft length. While these inlet ducts reduce the radar cross section, they also reduce the stability margin of the propulsion system. The more aggressive inlet ducts result in increased turning which in turn produces secondary flow within the duct.There are several design challenges posed by these serpentine inlet ducts. In many cases, the flow from the inlet duct is not of an acceptable flow quality for the engine airflow. The flow must be controlled or corrected to improve the flow quality entering the turbine engine. Several attempts to control the flow within the duct and improve exit flow uniformity have been demonstrated and will be discussed in the literature review. Additionally, the secondary flow generated within the serpentine duct will be more severe at off-design conditions for these more aggressive serpentine ducts.

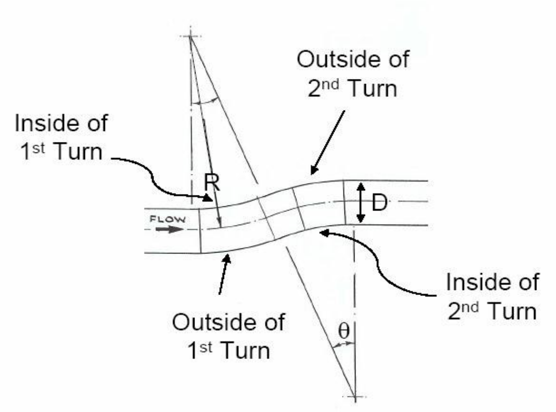


Fig 1
Serpentine-Duct with Common Terminology

1.3 CFD Analysis:

Computers have been used to solve fluid flow problems for many years. Numerous programs have been written to solve either specific problems, or specific classes of problems. From the mid-1970's, the complex mathematics required to generalize the algorithms began to be understood, and general purpose CFD solvers were developed. These began to appear in the early 1980's and required what were then very powerful computers, as well as an in-depth knowledge of fluid dynamics, and large amounts of time to set up simulations. Consequently, CFD was a tool used almost exclusively in research.Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models, have made the process of creating a CFD model and analysing results much less labour intensive, reducing time and, hence, cost. Advanced solvers contain algorithms that enable robust solutions of the flow field in a reasonable time.

As a result of these factors, Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world. CFD provides a cost-effective and accurate alternative to scale model testing, with variations on the simulation being performed quickly, offering obvious advantages.

2. THE MATHEMATICS OF CFD:

The set of equations that describe the processes of momentum, heat and mass transfer are known as the Navier-Stokes equations. These partial differential equations were derived in the early nineteenth century and have no known general analytical solution but can be discretized and solved numerically.

Equations describing other processes, such as combustion, can also be solved in conjunction with the Navier-Stokes equations. Often, an approximating model is used to derive these additional equations, turbulence models being a particularly important example. There are a number of different solution methods that are used in CFD codes. The most common, and the one on which CFX is based, is known as the finite volume technique.

In this technique, the region of interest is divided into small sub-regions, called control volumes. The equations are discretized and solved iteratively for each control volume. As a result, an approximation of the value of each variable at specific points throughout the domain can be obtained. In this way, one derives a full picture of the behaviour of the flow.

2.1 CFD Methodology

CFD can be used to determine the performance of a component at the design stage, or it can be used to analyse difficulties with an existing component and lead to its improved design. The first step is to identify the region of interest: The geometry of the region of interest is then defined. If the geometry already exists in CAD, it can be imported directly. The mesh is then created. After importing the mesh into the pre-processor, other elements of the simulation including the boundary conditions (inlets, outlets, and so on) and fluid properties are defined.

The flow solver is run to produce a file of results that contains the variation of velocity, pressure and any other variables throughout the region of interest. The results can be visualized and can provide the engineer an understanding of the behaviour of the fluid throughout the region of interest. This can lead to design modifications that can be tested by changing the geometry of the CFD model and seeing the effect.

3.0 DESIGN OF SERPENTINE INTAKECATIA V5R17:

CATIA is one of the leading design software. CATIA can be applied to a wide variety of industries, from aerospace and defence, automotive, and industrial equipment, to high tech, shipbuilding, consumer goods, plant design, consumer packaged goods, life sciences, architecture and construction, process power and petroleum, and services. We used Mechanical design feature for design the serpentine intake.

The design of serpentine intake is done by using CATIAV5R17 software with specifications.

The design is consists of five sections including two opposite bend sections with some angle to form shock in the intake.

3.1 Design

1:

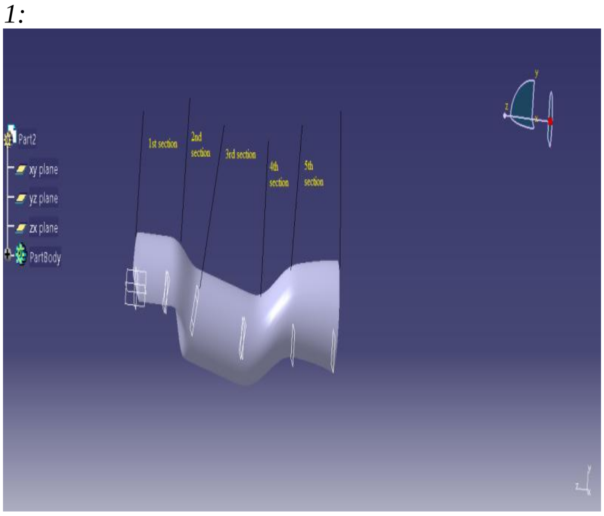


Fig 2
Side view of serpentine intake design 1

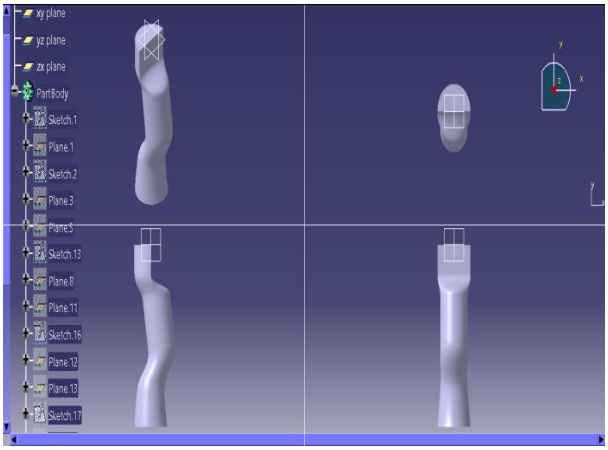


Fig 3
Various views of serpentine intake design 1

3.2 Dimensions

This diagram contains five section. The dimension of each section is listed below.

1st section:

Length- 30mm
Inlet semicircle
Diameter- 30mm

2nd section:

Angle – 10degree
Length -35mm

3rd section:

Length -50mm
Diameter -30mm

4th section:

Angle - 10degree
Length – 50mm
Diameter – 30mm

5th section:

Length-40 mm
Outlet diameter-30mm

For design 2:

4th section:

Angle – 20 degree
Length – 50mm
Diameter – 30mm

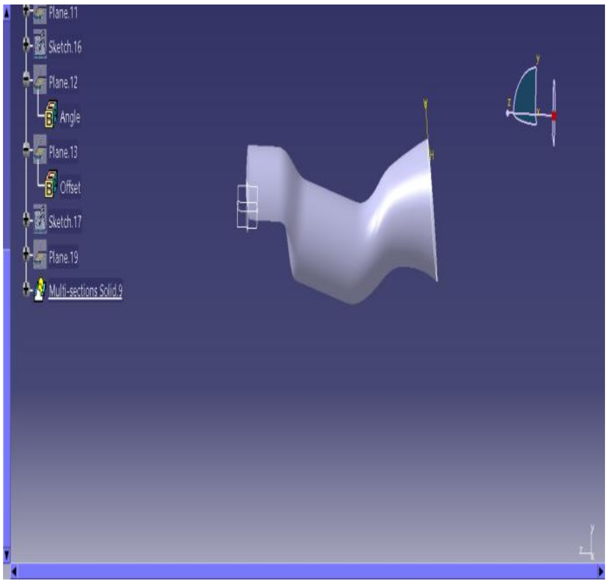


Fig 4
Side view of serpentine intake design 2

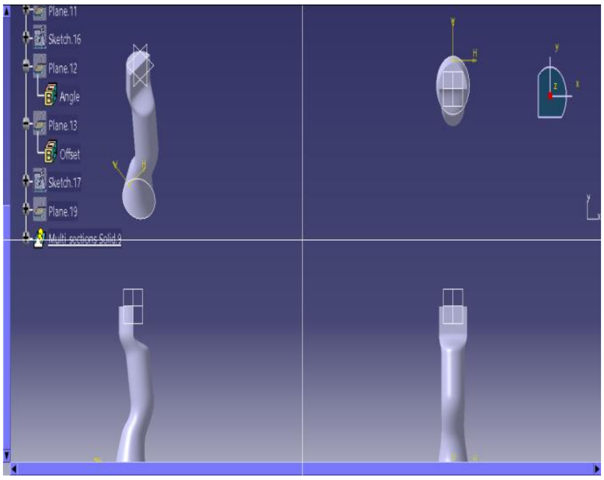


Fig 5
Various views of serpentine intake design 1

4. ANALYSIS IN ANSYS

4.1 Geomentry:

Design 1:

The serpentine design which is drown using CATIA V5R17 is imported in ansys.

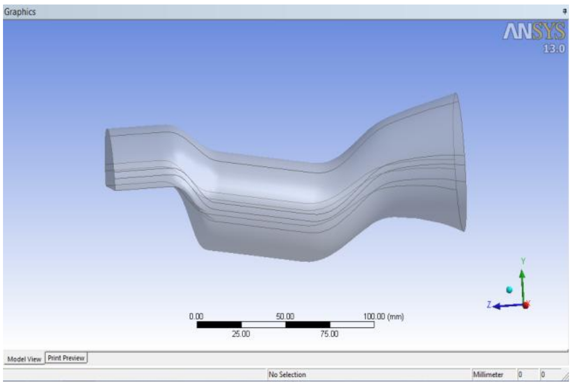


Fig 6
Imported geometry of design 1 in ANSYS

The above figure shows the imported geometry in ansys with freezed condition. It is ready for further setups in analysis.

Design 2:

The serpentine design which is drown using CATIA V5R17 is imported in ansys.

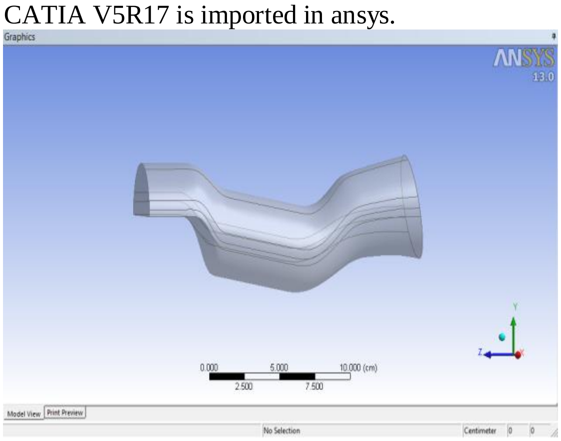


Fig 7
Imported geometry of design 2 in ANSYS

The above figure shows the imported geometry on ansys with freezed condition. It is ready for further setups in analysis.

4.2 Mesh:

The Model cell in the Mechanical application analysis systems or the Mechanical Model component system is associated with the Model branch in the Mechanical application and affects the definition of the geometry, coordinate systems, connections and mesh branches of the model definition

Grid development:

Grid quality is often an issue which is easily overlooked in computational Analysis and its consequences greatly undermined. Grid quality can significantly affect the results and lead to erroneous conclusions as clearly pointed by Logan and Nitta24. They pointed out that ideally a solution should be grid independent however this may never be perfectly attainable. Grid convergence simply refers to the ability of the grid to least interfere with the solution. It may be stated that this does not assure the trustworthiness of the solution as there are many other issues to CFD validation which are foreign to grid convergence. The simplest test for grid convergence is the usage of three different meshes with varying quality starting fromcoarse to fine.Grid convergence is assumed if there is a fair agreement between the results from the different meshes. For the inlet duct geometry, three individual meshes with varying mesh quality are generated. The node density among these grids varies with the ratio of 1:2:4 implying the medium quality having twice as many nodes as the coarse one and the fine grid having four times as many as the coarse grid as depicted.

For design 1:

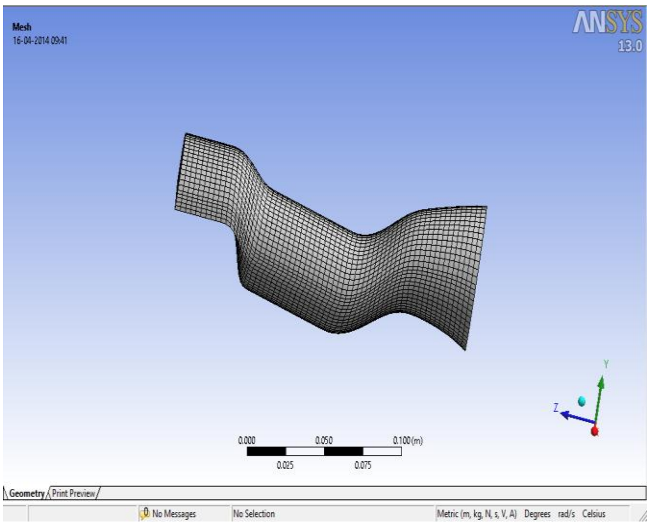


Fig.8
Geometry with fine mesh of design 1

For our design fine grid method is used for develop the mesh. Because it doesn't have complex structure in it. We don't need to increase the elements in the inlet and outlet face of intake.

By using the ANSYS CFX software the mesh was generated. The number of elements are 21350 and number of nodes are 23848.

For design 2:

For design 2 to develop mesh the same procedure which is followed in design1 is used.

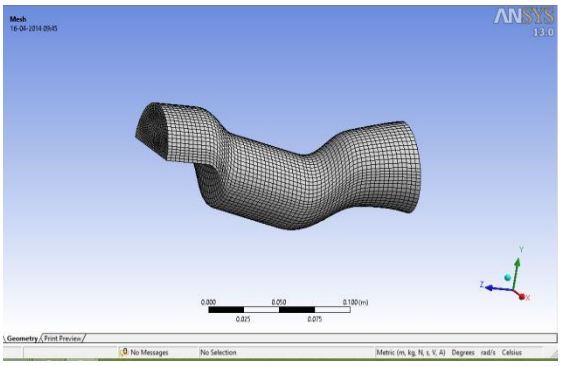


Fig.9
Geometry with Fine mesh of design 2

By using the ANSYS CFX software the mesh was generated. The number of elements are 20540 and number of nodes are 22846.

4.3 Setup:

Use the Setup cell to launch the appropriate application for that system. You will define your loads, boundary conditions, and otherwise configure your analysis in the application. The data from the application will then be incorporated in the project in ANSYS Workbench, including connections between systems.

For our design we need assign the domain as fluid domain and we need to apply the boundary condition for various section.

4.4 Boundary Conditions:

To analyse the performance of serpentine intake in ANSYS CFX there is a need to give some boundary condition.

The portion at which the high velocity fluid entered is assigned as inlet, the flow entered at the speed of 1000m/s and the flow regime is considered as supersonic.

The portion at which the high pressure air dissipate is called outlet, the relative pressure is given as 1.2pa and the flow regime is considered as supersonic.

The domain is considered as fluid domain and the domain is stationary, the fluid entered is considered at a temperature of 25^0C.The model is considered as non-buoyant and morphology is assigned as continuous flow. This model is a laminar model. No-slip wall condition is applied for boundary wall condition.

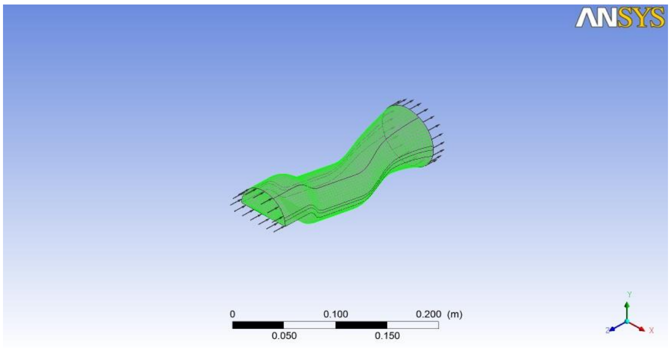


Fig 10
Boundary setup of geometry of design 1

For design 2:

For design 2 the similar boundary condition which is used in the analysis of design 1 is followed.

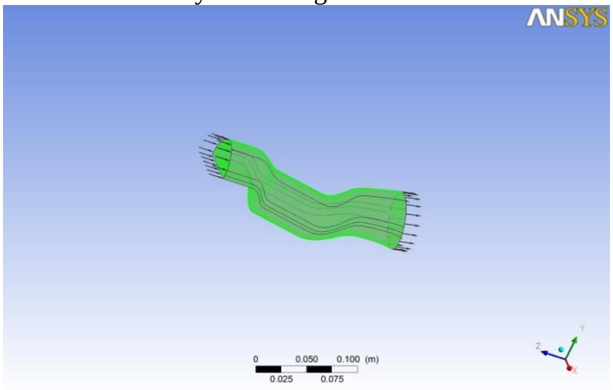


Fig 11
Boundary setup of geometry of design 2

4.5 Solver:

CFX-Solver Manager is a graphical user interface that enables you to set attributes for your CFD calculation, control the CFX-Solver interactively, and view information about the emerging solution. As an alternative to using the CFX-Solver Manager interface, you can also operate CFX-Solver from the command line, which is particularly useful for batch mode operations.

CFX-Solver Manager is an interface that displays a variety of results as outlined below. It is generally used to view the plotted data during problem solving.

For design 1:

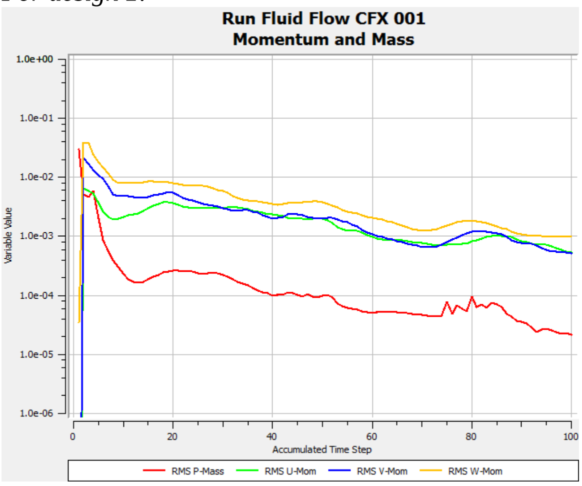


Fig 12
Residuals for baseline simulation for design 1

For design 2:

The same procedure which is used for design 1 is followed.

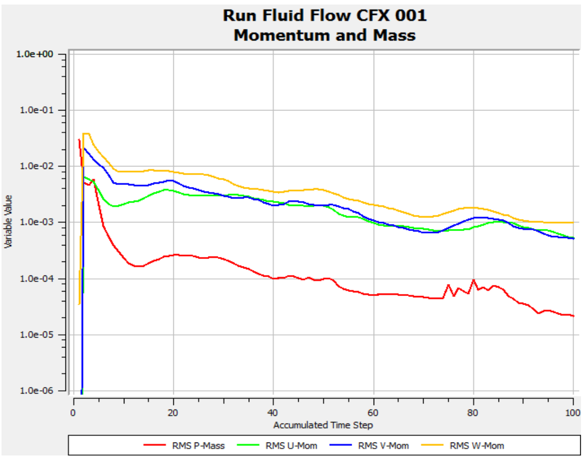


Fig 13
Residuals for baseline simulation for design 2

5.0 RESULTS

There is a various methods are available to display the analysis of serpentine intake.
We choose two methods to display the results.
Those are,

- Contour method
- XY Plot

5.1 Contour method:

ANSYS CFX allows you to plot contour lines or profiles superimposed on the physical domain. Contour lines are lines of constant magnitude for a selected variable (isotherms, isobars, etc.). A profile plot draws these contours projected off the surface along a reference vector by an amount proportional to the value of the plotted variable at each point on the surface.

The options mentioned in the procedure above include drawing colour-filled contours/profiles (instead of the default line contours/profiles), specifying a range of values to be contoured or profiled, including portions of the mesh in the contour or profile display, choosing node or cell values for display, and storing the contour or profile plot settings.

Pressure contour:
For design 1:

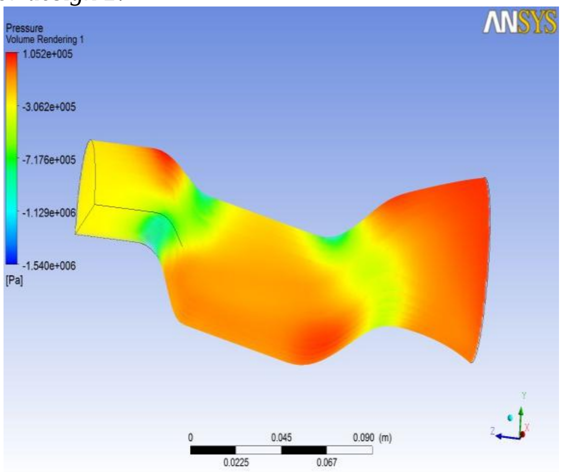


Fig 14
Pressure contour of design 1

For design 2:

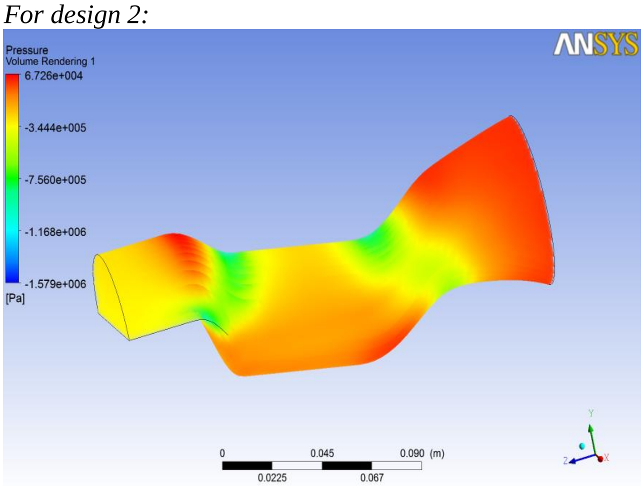
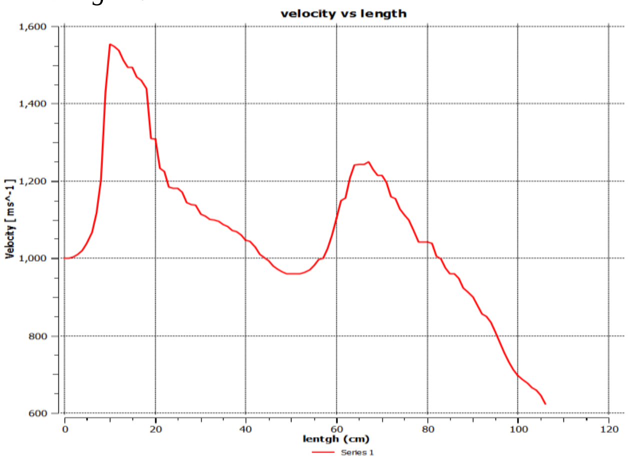


Fig 15
Pressure contour of design 2

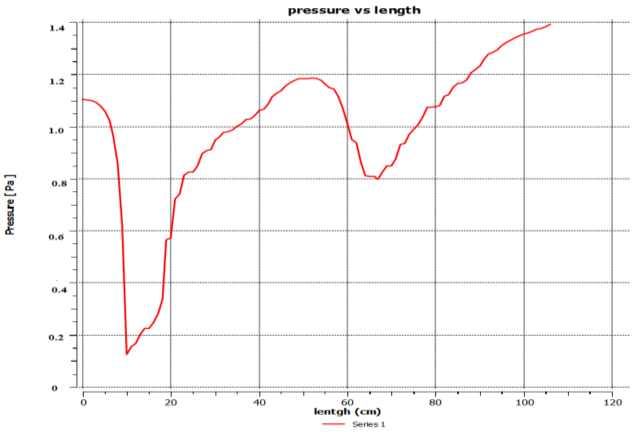
5.2 XY Plot:

ANSYS CFX allows you to plot the results in the XY Plot. It consists of two axis of X and Y.We used X axis for denote the sections of serpentine intakethrough length and Y axis to denote change in velocity of the serpentine intake.

For design 1:

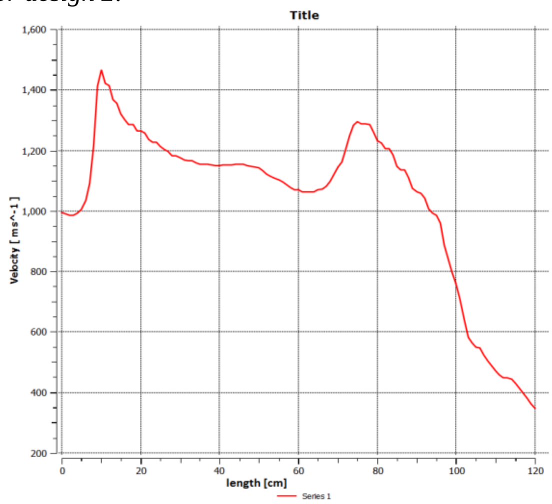


Graph 1
Velocity vs length of the intake design 1

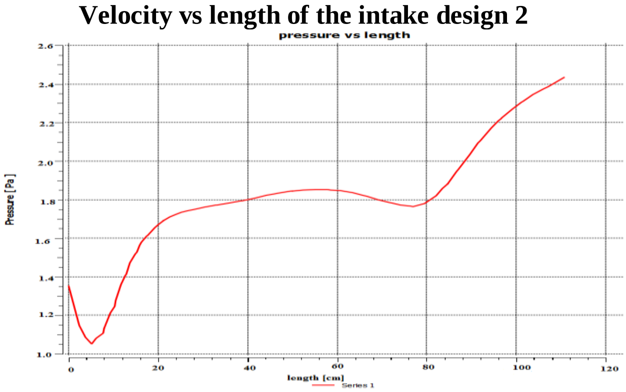


Graph 2
Pressure vs length of the intake for design 1

For design 2:



Graph 3
Velocity vs length of the intake design 2



Graph 4
Pressure vs length of the intake for design 2

6.0 CONCLUSION

From this project we can understand that there is a possibility of decrease in velocity of incoming flow to engine intake without the usage of intake spike. The results shows that the variation in the incoming flow velocity/pressure for two kind of designs with different curvature angle. The ANSYS CFX tool is used for finding the variations in the velocity and pressure of the incoming flow in the serpentine intake. For design 1 the curvature angles are 10/10 degrees to incoming velocity of the flow is decreased from 1000m/sec to 620m/sec and the pressure is increased from 1.2 Pa to 1.5 Pa. For design the curvature angles are 10/20 degrees to incoming flow velocity is decreased from 1000 m/sec to 350 m/sec and the static pressure is increased from 1.2 Pa to 2.3 Pa. By varying the curvature ratio of the section in the serpentine we achieved the significant changes in pressure and velocity of the incoming flow.

7.0 REFERENCE

1. Anderson, B.H., and Gibb, J., 1992, “Study on Vortex Generator Flow Control for the Management of Inlet Distortion,“ AIAA/SAE/ASME/ASEE 28th Joint Propulsion
2. Conference and Exhibit, AIAA-92-3177.
3. Bansod, P., and Bradshaw, P., 1972, “The Flow in S-shaped Ducts,” The Aeronautical Quarterly, Volume 23.
4. Guo, R.W., and Seddon, J., 1983, “The Swirl in an S-Duct Inlet of Typical Air Intake Proportions,” Aeronautical Quarterly
5. Jaw, L.C., Cousins, W.T., Wu, D.N., and Bryg, D.J., 2001, “Design and Test of a Semi-
6. Passive Flow Control Device for Inlet Distortion Suppression,” Journal of Turbo machinery,Volume 123, pp. 9- 13.
7. Angela C. Rabe August 1, 2003,”Effectiveness of a Serpentine Inlet Duct Flow Control Scheme at Design and Off-Design Simulated Flight Conditions”.
8. Abhinav Kumar August, 2007 “Flow control optimization in a jet engineserpentine inlet duct”Yu, S.C., & Chan, W.K. (October 1996).
9. Effect of a central straight on an s-shaped diffusing duct. Aeronautical Journal, Paper No. 2182.
10. Sullivan, J.P., Murthy, S.N.B., Davis. R., & Hong, S. (1982). *S-shaped duct flows*. Office of Naval Research Contract Number N-78-C-0710
11. Gerolymos, G. A., Sauret, E., and Vallet, I., “Oblique-Shock-Wave Boundary-Layer Interaction Using Near-Wall Reynolds-Stress Models,” AIAA Journal, Vol. 42, No. 6, June 2004.